PMOD is a software
FOR RESEARCH USE ONLY (RUO)
and must not be used for diagnosis or treatment of patients.
This page is intentionally left blank.
# Table of Contents

1. **PMOD Geometric Models and Simulation Tool (PGEM)**
   1.1 User Interface .................................................................................. 5
   1.2 Configuration Settings ..................................................................... 8
   1.3 OpenFoam CFD Server ...................................................................... 9
   1.4 Additional Documentation ................................................................. 13

2. **DTI Tractography**
   2.1 Workflow Organization .................................................................... 14
   2.2 Loading the Diffusion Tensor Images ................................................. 14
   2.3 Defining VOIs for Seeding and Imposing Restrictions ......................... 15
   2.4 Calculating the Fiber Tracks ............................................................... 18
   2.5 Visualizing the Fiber Tracks in 3D ...................................................... 21
   2.6 DTI Report ....................................................................................... 24
   2.7 Evaluating Diffusion Images along Tracks .......................................... 26
   2.8 Saving the Tracks ............................................................................. 26
   2.9 Protocol File ..................................................................................... 27

3. **4D Flow**
   3.1 Workflow Organization .................................................................... 28
   3.2 Loading the 4D Flow Images ............................................................. 28
   3.3 Entering the Maximal Flow Value ..................................................... 30
   3.4 Defining VOIs for Seeding and Imposing Restrictions ......................... 30
   3.5 Calculating the Streamlines/Pathlines ............................................... 33
   3.6 Visualizing the Streamlines/Pathlines in 3D ....................................... 36
   3.7 4D Flow Report ............................................................................... 37
   3.8 Creating VOIs along the Streamlines/Pathlines .................................. 40
   3.9 Saving the Streamlines/Pathlines ...................................................... 41
   3.10 Protocol File .................................................................................... 41

4. **Generation of Dynamic PET Phantom Images**
   4.1 External Preparations ....................................................................... 42
   4.1.1 Phantom VOIs ................................................................................ 42
   4.1.2 Phantom Dynamic TACs ................................................................. 42
   4.2 Dynamic PET Image Creation ............................................................. 43
   4.2.1 Anatomical Model Definition ......................................................... 43
   4.2.2 VOI Addition .................................................................................. 45
   4.2.3 Anatomical Model ......................................................................... 46
   4.2.4 Phantom Creation ......................................................................... 46

5. **Geometric Model Construction and CFD Simulation**  

---

(C) 1996-2021 PMOD Geometric Models and Simulation (PGEM)

---
5.1 Geometric Model Construction ................................................................. 49
  5.1.1 Workflow Organization ........................................................................... 49
  5.1.2 Start of the Model Definition ................................................................. 49
  5.1.3 Time Definition ........................................................................................ 50
  5.1.4 Image Definition ....................................................................................... 52
  5.1.5 VOI Definition .......................................................................................... 55
  5.1.6 VOI Addition ............................................................................................ 56
  5.1.7 Anatomical Model ..................................................................................... 60
    5.1.7.1 Tree Organization of the Anatomical Model ........................................... 61
    5.1.7.2 Components of the Model Folder in PMOD ............................................. 64
    5.1.7.3 Definition of the Characteristics for the Model’s Elements ...................... 67
    5.1.7.4 Definition of the Properties for the Model’s Elements ............................. 69
    5.1.7.5 Interactive Exploration of the Model .................................................... 70

5.2 CFD Simulation and Implementation in PGEM ........................................... 72
  5.2.1 Create CFD Case from the Geometric Model .......................................... 74
    5.2.1.1 Geometry Selection in the Model ........................................................... 74
    5.2.1.2 Surface Mesh Creation .......................................................................... 75
      5.2.1.2.1 Cutting Planes Definition with 3D ......................................................... 77
    5.2.1.3 Volume Mesh Creation .......................................................................... 80
    5.2.1.4 Boundary Condition Settings ............................................................... 82
    5.2.1.5 Solution Control .................................................................................... 84
  5.2.2 Create CFD Case from a Surface Mesh .................................................. 85
  5.2.3 Create CFD Case from a Volume Mesh ................................................... 86
  5.2.4 Create CFD case from an Existing Case ................................................ 88
  5.2.5 Management of the OpenFoam Remote Cases ....................................... 89
  5.2.6 Management of the OpenFoam Local Cases .......................................... 92
    5.2.6.1 Conversion of CFD Results to Images ................................................... 94
    5.2.6.2 Visualization of the CFD Results ............................................................ 95
      5.2.6.2.1 As Polymesh ................................................................................. 95
      5.2.6.2.2 As Volume Rendering .............................................................. 99

6. References ......................................................................................... 102

7. PMOD Copyright Notice ............................................................. 103

Index .................................................................................................. 105
1 **PMOD Geometric Models and Simulation Tool (PGEM)**

The PGEM tool supports various types of analyses and simulations based on anatomical data. Particularly, the following tasks are supported:

- **DTI Tractography**: Diffusion tensors resulting from the analysis of DWI MR data can be imported and three-dimensional tracts generated. They may be visualized in 3D scenes, used for statistics on the diffusion maps, and exported in vtk format for use in external programs.

- **4D Flow Visualization**: MR sequences are able to measure the instantaneous flow vector in vessels over time. In a similar analysis as in tractography, this information can be used for generating streamlines which visualize flow direction and velocity.

- **Generation of Dynamic PET images**: Based on an anatomical atlas and a time-activity curve for each atlas structure, a synthetic dynamic PET image can be generated with a certain noise level. Such data can be used for the evaluation of parametric mapping and segmentation approaches.

- **Construction of Vessel Structures and Computational Fluid Dynamics (CFD) simulation**: In a first step vessel structures can be segmented from anatomical images and converted into a geometric model. In a second step, flow through the vessel is simulated in an external CFD software. The resulting flows can subsequently be retrieved and analyzed.

1.1 User Interface

**Starting the PGEM Tool**

The PGEM tool is started with the **Anatomy** button from the PMOD Toolbox:

The user interface of PGEM consists of five pages which can be selected by tabs:

1. **Load Images** page: This page is only available when using PMOD databases.
2. **VOIs & Tracking** page: Main page for DTI and 4D Flow analysis, as well as the VOI definition for the anatomical models.
3. **Models** page: Page for geometric model definition and calling CFD modeling.
4. **Phantoms** page: Page for using a geometric model for the generation of synthetic images.
5. **3D** page: Page for results are visualized with 3D rendering.
Workflows
The main workflows share the pages. Color coding is therefore applied for highlighting the elements belonging to a certain workflow: red for DTI, blue for 4D Flow, green for anatomical model definition and CFD simulation.

Taskbar
The taskbar to the right of the application window provides shortcut buttons for frequent tasks. Please note the tooltips when hovering over the buttons.

- **Load images.** The arrow below the loading button allows changing the data format.
- **Clear all data.**
- **Save results.** A dialog window appears for selecting the relevant parts and defining the output format.
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DTI</strong></td>
<td>Start DTI workflow by loading tensor images.</td>
</tr>
<tr>
<td><strong>4DF</strong></td>
<td>Start 4D Flow workflow by loading MR images with 4D flow information.</td>
</tr>
<tr>
<td><strong>IML</strong></td>
<td>Start model construction workflow by loading anatomical images.</td>
</tr>
<tr>
<td>VOIs</td>
<td>Create a new anatomic model.</td>
</tr>
<tr>
<td>VOIs</td>
<td>Open the <strong>VOIs</strong> panel on the <strong>VOIS &amp; Tracking</strong> page.</td>
</tr>
<tr>
<td><strong>4DF</strong></td>
<td>Load VOIs from a file and add them to the current model.</td>
</tr>
<tr>
<td><strong>4DF</strong></td>
<td>Open the manager of the remote OpenFOAM cases.</td>
</tr>
<tr>
<td><strong>4DF</strong></td>
<td>Open the manager of the local OpenFOAM cases.</td>
</tr>
<tr>
<td><strong>4DF</strong></td>
<td>Allows viewing linked CFD case.</td>
</tr>
<tr>
<td><strong>4DF</strong></td>
<td>Rendering the selected model structures in 3D.</td>
</tr>
<tr>
<td><strong>4DF</strong></td>
<td>Aggregation of VOI statistics results for 4D Flow.</td>
</tr>
</tbody>
</table>
Hidden Controls
In some places options are hidden to save screen space. This is indicated by a blue up-arrow as in the example below.

When the button is activated, the area expands, showing all the options. It can be collapsed again with the green down-arrow.

1.2 Configuration Settings

The PGEM tool can be configured according to user preference using the icon next to the Flow menu button.

The Structural models path defines, where the created models will be stored.

In the CFD Modeling section the OpenFOAM CFD server can be configured. Per default, PMOD comes configured with an OpenFoam server hosted on a PMOD machine (anatomy.pmod.com:5998). However, this open foam server should only be used for tests, and not for productive data processing. We recommend users to set up their own server, which is easy to do and free of charge. A description of the OpenFOAM installation on Ubuntu and the PMOD configuration is available in the following section.
The **Case path (local)** allows defining the directory where the CFD cases will be downloaded from the OpenFOAM server. Similarly, a **Backup path** can be configured.

The **3D background color** settings serve for predefining the background in the scenes created by the different workflows.

The **Start on page** should be configured such that the most frequently used page is opened when starting PGEM up.

### 1.3 OpenFoam CFD Server

**OpenFoam Installation on Ubuntu**

OpenFoam 5.0 installation for Ubuntu is described at: https://openfoam.org/download/5-0-ubuntu/

Installation steps for Ubuntu 16.04 and OpenFoam 5.0:

1. Add OpenFoam repository
   ```bash
   sudo add-apt-repository http://dl.openfoam.org/ubuntu
   sudo sh -c "wget -O - http://dl.openfoam.org/gpg.key | apt-key add -"
   ``
2. Update package list
   ```bash
   sudo apt-get update
   ``
3. Install OpenFoam and Paraview
   ```bash
   sudo apt-get -y install openfoam5
   ``
4. Edit shell configuration, open .bashrc in editor
   ```bash
   gedit ~/.bashrc
   ``
   and add at the end of file:
   ```bash
   source /opt/openfoam5/etc/bashrc
   ``

**Pmod Configuration for OpenFoam**

Once PMOD is properly installed, the OpenFoam CFD server can be easily configured performing the following steps:

1. Open the main configuration interface using the **Config** button available on the **PMOD Toolbox**.
2. Select the **DATABASE** tab and **Add new source** as illustrated below:

   ![Database Configuration Interface](image)

3. Activate the **Create Database** button and configure the **Path** to OpenFoam:
Note that path should point to OpenFoam main folder e.g. /opt/openfoam5/

4. Switch the radio button from Use Direct Connection to Use Transaction Server:

5. Configure the properties of the transaction server. The Secure box is for enabling secure communication. This mode should be used if the communication is not confined within the institution. Otherwise it will slow down the communication speed unnecessarily.

6. An important property is the IP Port for the communication. It must be a unique number not used by any other transaction server or other process. The default port is 5202 if no other transaction server was configured during the PMOD installation.

7. Another important property is the IP address. It must contain the address of the host in which the servers are running, so typically the system on which the configuration is performed. For this system the IP address can be obtained by activating the Set Local Host button. Note that entering "localhost" in the HOST area will NOT work!

8. Check the box Stand alone and activate the Save Starting Script button. A dialog window appears which shows the contents of the created script. The example below shows the result for Ubuntu system:

9. Select Yes to save the starting script with the specified Name in the specified subdirectory of the PMOD installation directory:
10. Save the PMOD configuration with the OK button and exit PMOD.

11. Start the Pmod Transaction Server by double-clicking the script file. Please note that Transaction Server doesn’t need a WIBU Key to run, it can be unplugged.

**Pmod configuration for OpenFoam (client side)**

Start the PGEM local configuration to configure the OpenFoam PMOD client. The steps for an appropriate configuration are as follows:

1. In the **CFD Modeling** section enable the **Server** radio button. Define the server by the **host** name or **IP** address and set the **Port** number used in the openFoam transaction server configuration as indicated below:

2. Set the number of cores on the server to be used when solving CFD cases with the OpenFoam:
3. Define the local path (client side) to the cases folder:

Note that cases downloaded from the server will be stored on the location specified on the Case path (local).

4. Finally, activate the **Echo and Manage OpenFOAM cases** button to verify the configuration:
1.4 Additional Documentation

The PGEM tool is based on a range of functionalities from the PMOD environment which are not explained in this documentation. Please refer to the

- **PMOD Base Functionality Guide** for information related to data loading, image viewing, volume-of-interest (VOI) definition and the
- **PMOD 3D Rendering Tool Users Guide** for rendering-related information.
2 DTI Tractography

This chapter describes the workflow for loading diffusion tensor data and performing tractography.

2.1 Workflow Organization

The DTI workflow starts on the VOIs & Tracking page, which has 4 sub-pages. As illustrated below, the sub-pages can be switched with the selection in the upper right or the adjacent arrows.

For DTI, only IMAGE, VOIs and TRACKING are relevant. Therefore, FLOW XYZ is disabled as soon as the DTI tensor images have been loaded.

On each page, the user has to perform an operation or configuration, and then activate the red action button in the lower right for proceeding. Correspondingly, an operation is performed and the result shown on the next page. After leaving the TRACKING page, the resulting fiber tracks are visualized on the 3D page.

In order to test a different tracking configuration, the user has to return to a prior stage by selecting the VOIs & Tracking page again, choosing the appropriate sub-page, and then proceeding from there as with the initial workflow by the red action buttons.

2.2 Loading the Diffusion Tensor Images

Most conveniently, the diffusion tensor images are loaded using the button in the taskbar to the right. As an alternative, the Tensor radio button can be set on the IMAGE page and the images loaded with the button indicated below.

As a result of this loading, the workflow type is defined as "DTI tractography" and the internal program switches are configured accordingly. Particularly, the action button in the lower right is set to red color and indicates that the next step continues on the VOIs sub-page.
2.3 Defining VOIs for Seeding and Imposing Restrictions

When arriving at the VOIs sub-page a fusion image is shown to the left which is composed of the Tensor image and the FA (Fractional Anisotropy) map. Note the two panels A and B in the upper right which correspond to these images, and the slider for mixing the respective contributions in the fusion image.
In the lower right, the A and B images can be redefined. The mean diffusivity (MD), the DTI tensor as well as a masked version of the FA image are available for selection.

VOI Definition

In PGEM, tractography is based on VOIs which can play different roles:

- **SEED** VOIs: Tractography is started from the pixels in seed VOIs, following the local vector directions.
- **END** VOIs: Only tracks ending in this VOI will be used. To avoid excessive pruning of tracks, an END VOI should have sufficient volume.
- **WAY** VOIs: Only tracks passing through the WAY VOI will be used.
- **EXCLUDE** VOIs: Tracks passing through an EXCLUDE VOI will be removed.
- **NEUTRAL** VOIs: Such VOIs have no impact on tractography. Setting VOIs to NEUTRAL allows assessing the effect of END, WAY and EXCLUDE VOIs without the need of removing them.
- **TARGET**: Is representing a VOI the track should end with.

VOIs can be created with the usual PMOD VOI tools. When creating a new VOI, a dialog window pops up which requires defining the appropriate Tracking property. Later on, this property can be changed using the VOI properties functionality as illustrated below.
In the following sections two VOIs will be used: a **SEED VOI** in the left corpus callosum created using the paint tool, and an elliptic VOI which will be used as a **WAY VOI**. The illustration below shows the VOIs on top of the FA image.

After the VOIs have been defined, please proceed with the **Tracking** button in the lower right.

Note the check box **Vorticity&Helicity**. If it is checked, PGEM calculates parametric maps called **Vorticity Amplitude** and **Absolute Helicity**, respectively. Vorticity is a measure of rotation, whereas helicity relates to the knottedness in fluids.

**Tensor Inspection**

The loaded tensor data can be inspected with the **QC** button. It shows a window with the projection of the principal diffusion direction vectors together with a color-coded direction image. Red indicates diffusion along the x-axis (left-right), green diffusion along the y-axis (posterior-anterior), and blue diffusion along the z-axis (inferior-superior).
2.4 Calculating the Fiber Tracks

When arriving at the TRACKING page, the image area shows the FA image (A), which highlights areas of high diffusion anisotropy. However, a second image (B) is available for fusion. To see both images, the fusion slider has to be shifted to the right.

In the example below, the FA image is combined with the MD image, which highlights the CSF space. Also shown in the overlay are the VOIs.
Tracking Configuration

The area in the lower right serves for defining the tracking algorithm and its parameters. DTI fiber tracking algorithms can be divided into **DETERMINISTIC** and probabilistic methods. PMOD supports the following probabilistic fiber tracking methods: **PICO** for DTI with pregenerated LUT tables (from the PXMOD module) and Bayesian tracking (on simplified tensors, ball and stick model) directly on DWI images.

The **DETERMINISTIC** method initiates fiber trajectories from user-defined voxels. For example, in the fiber assignment by continuous tracking algorithm (FACT) the fiber trajectories, also known as “streamlines”, follow the primary eigenvector from voxel to voxel in 3 dimensions. When the fiber trajectory reaches the edge of the voxel, the direction of the trajectory is changed to match the primary eigenvector of the next voxel. Constraints on the maximum turning angle of the streamline between voxels and on the minimum FA within a voxel for propagation of the streamline can be applied to contain the fiber tracks to regions of the brain where the diffusion tensor model realistically represents the white matter pathways[4].

The deterministic streamline fiber tracking is influenced by the noise, subject movement, and distortion from imaging artifacts which produce uncertainty in the orientation of the diffusion ellipsoid[4].

The probabilistic fiber tracking methods compensate the drawback incorporating the expected uncertainty into the tracking algorithm and can be used to produce a connectivity metric for each voxel. Probabilistic fiber tracking techniques tend to disperse trajectories more than deterministic methods and have the potential to delineate a greater portion of a white matter tract. However, the accuracy of these probabilistic methods is still limited by the information contained in the diffusion tensor and the method of constructing the probability density function[4].

For each tracking algorithm three methods are supported and can be selected in the **Method** list:

1. **FACT**: Similar to the FACT algorithm proposed by Mori et al [1], this method follows the local fiber orientation in each voxel. No interpolation is used.

2. **EULER**: Tracking proceeds using a fixed step size along the local fiber orientation [2]. With nearest-neighbor interpolation, this method may be very similar to FACT, except that the step
size is fixed, whereas FACT steps extend to the boundary of the next voxel (distance variable depending on the entry and exit points to the voxel).

3. **4th order Runge-Kutta** method [2]. The step size is fixed, however the eventual direction of the step is determined by taking and averaging a series of partial steps. Interpolation determines how the fiber direction is determined at a given point in space.

The **Interpolation** setting is not relevant for FACT. For **EULER** and **4th Runge-Kutta** the choices are:

1. **NN**: Nearest-neighbor interpolation using the local voxel data directly.
2. **NC**: Probabilistic nearest-neighbor interpolation, similar to the method proposed by Behrens et al [3]. The data is not interpolated, but at each point one of the 8 voxels neighbors is randomly chosen. The probability of choosing a particular voxel is based on how close the point is to the center of that voxel.
3. **TRILINEAR**: Linear interpolation of the vector components using the 8 neighbors.

Further parameters and options determine how tracking proceeds and when it ends:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Anisotropy</strong></td>
<td>Track ends if the fractional anisotropy (which is in the range [0,1]) drops below this threshold.</td>
</tr>
<tr>
<td><strong>Step size</strong></td>
<td>Distance between points on the track. Not used for FACT as it does not apply interpolation.</td>
</tr>
<tr>
<td><strong>Min length</strong></td>
<td>If checked, tracks shorter than the minimal length specified are discarded.</td>
</tr>
<tr>
<td><strong>Max curvature</strong></td>
<td>If checked, tracks are terminated in case the curvature exceeds the specified angle within the length specified. If length = 0.0, the curvature is checked at each step.</td>
</tr>
</tbody>
</table>
| **Exclusion VOI** | Defines the behavior, when a track enters an EXCLUDE VOI:  
|                  | **DISCARD** the whole track, or **CLIP** it at the entry point. |
| **Colour by**   | The track lines can be functionally colored as follows:  
|                 | **SEED**: Using the color of the **SEED** VOI.  
|                 | **END VOI**: Using the color of the **END** VOI.  
|                 | **DIRECTION**: Each segment of the track has a color combined from its direction cosines values: Red, Green, and Blue from direction cosines x, y and z respectively.  
|                 | **ANISOTROPY MAP**: Using the FA value along the track for coloring. |

After configuring tracking, please proceed with the **Fiber Tracking (3D)** button. The tracks are calculated and visualized on the **3D** page.

**Track Masking**

An additional means for restricting the generated tracks is by checking the **Mask fibers** box and defining a mask. The elements in

![Mask fibers](image)

have the following functionality:

- Generate a mask from the FA image by applying the specified threshold and include it in the image list for viewing.

![Generate mask](image)
Open the current mask in the VOI editor and convert it into a VOI. The VOI can be edited, and the mask finally updated from the VOI.

Select an external mask file.

References

2.5 Visualizing the Fiber Tracks in 3D

The 3D page initially shows the tracks together with the VOIs.

Note the elements in the Scene tree in the upper right. An objects can be selected in the tree its appearance changed in the lower part. Please refer to the PMOD 3D Tool Users Guide for details about the rendering.

As an example, FA image planes can be added by clicking at Planes in the tree, activating the Orthogonal button for the Add planes, opening the planes locator and finally selecting the FA image.
Examples with Different VOI Settings

The two VOIs described above will be used for illustrating the effect of various settings on the resulting tracks. The left corpus callosum VOI used for seeding, and the tracking algorithm is configured by the setting below.

The following results are obtained when changing the role of the ellipse VOI.
<table>
<thead>
<tr>
<th>Ellipse: NEUTRAL VOI</th>
<th>The scene shows all tracks found with the current algorithm from the seeding VOI.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ellipse: WAY VOI</td>
<td>By enforcing tracks to pass through the ellipse, the number of resulting tracks is decreased.</td>
</tr>
<tr>
<td>Ellipse: END VOI</td>
<td>Enforcing tracks to end within the ellipse VOI results in an empty scene, as none of the tracks fulfills this restriction.</td>
</tr>
</tbody>
</table>
### 2.6 DTI Report

The button on the 3D page composes a report as illustrated below:

<table>
<thead>
<tr>
<th>Ellipse: EXCLUDE VOI with CLIP setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Using the ellipse VOI for clipping results in all tracks being shown but clipped at the level of the ellipse.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Ellipse: EXCLUDE VOI with DISCARD setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>When discarding all tracks passing the ellipse only few tracks remain in the scene.</td>
</tr>
</tbody>
</table>
The report is organized on a single page: **Values.**

The report lists on the upper part the statistics of all defined VOIs in table format. For each VOI the **Type** is shown. The following statistics measures are displayed:

- **AVG FA[1/1]:** is the fractional anisotropy and is representing a summary measure of microstructural integrity. The FA is highly sensitive to microstructural changes, but it is less specific to the type of change [2].
  It is calculated for each VOI type. The table displays the maximum of all the VOI averages.

- **AVG. CURVATURE[deg]:** The DTI tracks allows visualizing the lines along which there is oxygen diffusion. Curvature is a measure of curvilinearity of neural connections.
  It is calculated only for the **NEUTRAL** VOI type in respect to each **SEED** VOI available on the VOIs list.
  The value shown in the table for each neutral VOI is representing the value of the normalized curvature of the set of tracks. The normalized curvature is calculated as the sum of the angles values between consecutive track's parts, divided by the number of angles.

- **AVG. TRACK LENGTH[mm]:** is representing the average length of the tracks which pass through a **SEED** VOI. This parameter is calculated for each **SEED** VOI available in the VOIs list.

**Note:** Helicity and vorticity parameters are estimated in the DTI workflow when the check box **Vorticity&Helicity** is enabled in the VOI definition step. The **AVG. HELICITY** and **AVG. VORTICITY** shown in the table are representing the average values estimated in each VOI available in the VOIs list.

The lower part of the report window shows a capture of the current 3D scene.

As usually, the report can be printed or captured in various formats via **Print report. Save results** serves for saving the numeric values, which can also be exported via **Copy to Clipboard.**
2.7 Evaluating Diffusion Images along Tracks

It may be of interest to evaluate diffusion parametric maps such as FA and MD images along the tracks. The solution offered by PGEM is to convert the fibers into VOIs, which can externally be used for statistics.

Please activate the Streamline VOIs button on the 3D page to convert the tracks into VOI outlines and open them in a VOI editor window.

Here they can be modified using the standard PMOD VOI tools, and saved for later usage.

2.8 Saving the Tracks

The generated tracks can be saved in vtk format for use in the standalone 3D tool, or for use with a third-party program. Please make sure that the 3D page is selected, as the saving functionality is page-specific. Then select the saving button in the taskbar to save the tracks.

Correspondingly, already existing vtk tracks can be loaded or appended as illustrated below.
2.9 Protocol File

The configuration of the DTI analysis can be saved with the Save Protocol button in the status line. The adjacent Load Protocol button allows retrieving the configuration and repeating the analysis.
3 4D Flow

The PGEM allows loading and analyzing data generated by 4D Flow MRI acquisitions. The workflow is very similar to the one described for DTI Tractography, and the applied tracking methods are exactly the same.

In 4D Flow analysis, the flow velocity vector takes the role of the principal tensor eigenvector in the DTI analysis. The result of the analysis are "streamlines", comparable to the fiber tracks of DTI tractography. Streamlines connect the velocity vectors at a given acquisition time, and give a general impression of the flow pattern at that time. As 4D Flow measurements acquire the velocity vectors at multiple times, e.g. at different phases of a heartbeat, streamlines can be calculated at every acquisition time. As a result, the change of the flow pattern over time can conveniently be visualized as a movie.

An alternative analysis is the flow tracking along time, connecting the velocity vectors of successive acquisition and thereby mimicking the trajectory a particle might trace during the acquisition. The result of such an analysis are called "pathlines". As of now, PGEM doesn't support the generation of pathlines, only streamlines.

3.1 Workflow Organization

The 4D Flow workflow starts on the VOIs & Tracking page, which has 4 sub-pages. As illustrated below, the sub-pages can be switched with the selection in the upper right or the adjacent arrows.

On each page, the user has to perform an operation or configuration, and then activate the blue action button in the lower right for proceeding. Correspondingly, an operation is performed and the result shown on the next page. After leaving the TRACKING page, the resulting streamlines are visualized on the 3D page.

In order to test a different tracking configuration, the user has to return to a prior stage by selecting the VOIs & Tracking page again, choosing the appropriate sub-page, and then proceeding from there as with the initial workflow by the blue action buttons.

3.2 Loading the 4D Flow Images

The 4D Flow analysis requires two data sets, "Magnitude" and a "Flow velocity", which are generated by MR reconstruction. The magnitude image shows anatomical detail and has arbitrary scaling, whereas the flow velocity image is a 3D vector field of the flow vector in real-world units.

Most conveniently, the 4D Flow images are loaded using the button in the taskbar to the right. Note that the flow magnitude and the flow vectors can be selected and loaded at once. PGEM figures out the roles of the two data sets based on their dimension. In the example below flow has been recorded at 19 different times during the heartbeat. The flow magnitude data set MAG therefore has a value of 19 in the nv column. The flow velocity in each point is a three-dimensional vector and therefore the FLOW data set requires three times as many volumes, namely 57.
As an alternative, the 4D Flow radio button can be set to **Magnitude** and the data loaded with the indicated button. Again, both the magnitude and velocity can be loaded at once, if the velocity vector information is in a single file.

In case each flow vector component is stored in a separate file, loading works differently. First load the magnitude image with the radio button **Magnitude**, then the three direction components radio button **Velocity [3]** as illustrated below. Note that the three series have to be selected and brought into the correct order (x, y, z) before loading them.

As a result of this loading, the workflow type is defined as "4D Flow" and the internal program switches are configured accordingly. Particularly, the action button in the lower right is set to blue color and indicates that the next step continues on the FLOW XYZ sub-page.

Please activate **Flow XYZ** to proceed.
3.3 Entering the Maximal Flow Value

The loaded images are shown on the FLOW XYZ page in a fusion arrangement. Note the fusion slider for mixing the contributions of the MAG and the FLOW images. The X, Y and Z radio buttons define, which velocity component of FLOW is displayed.

The VENC parameter values are required for converting the FLOW image values into real-word flow velocities. If they are not encoded in the loaded DICOM file, they have to be edited manually.

Please proceed with the VOIs button to the next page.

3.4 Defining VOIs for Seeding and Imposing Restrictions

When arriving at the VOIs sub-page a fusion image is shown to the left which is composed of the Mag and the FLOW MASK (Angio) images. Note the two panels A and B in the upper right which correspond to those images, and the slider for mixing the respective contributions in the fusion image.
In the lower right, the A and B images can be redefined.

The choices are
- the flow **VELOCITY** components in the different directions $X$, $Y$, $Z$,
- the **VELOCITY AMPLITUDE** (length of velocity vector),
- an image **ANGIO STATIC** resembling an angiographic image (calculated by summing $\sqrt{VELOCITY \times MAGNITUDE}$ over time),
- an image **ANGIO MULTIFRAME**
- a masked version of **ANGIO STATIC** at the 20% threshold level,
- the **MAG**
- the **INP:FLOW**

**VOI Definition**
The calculation of streamlines is based on VOIs which can play different roles:
- **SEED** VOIs: Streamlines tracking is started from the pixels in seed VOIs, following the local velocity vector directions.
- **END** VOIs: Only streamlines ending in this VOI will be used. To avoid excessive pruning of streamlines, an **END** VOI should have sufficient volume.
- **WAY** VOIs: Only streamlines passing through the **WAY** VOI will be used.
- **EXCLUDE** VOIs: Streamlines passing through an **EXCLUDE** VOI will be removed.
- **NEUTRAL** VOIs: Such VOIs have no impact on streamlines tracking. Setting VOIs to **NEUTRAL** allows assessing the effect of **END**, **WAY** and **EXCLUDE** VOIs without the need of removing them.
- **TARGET**: Is representing a VOI the track should end with.

VOIs can be created with the usual PMOD VOI tools. When creating a new VOI, a dialog window pops up which requires defining the appropriate Tracking property. Later on, this property can be changed using the **VOI properties** functionality as illustrated below.

Note the check boxes **Energy loss** and **Vorticity&Helicity**. If they are checked, PGEM calculates energy loss map and **Vorticity Amplitude** and **Absolute Helicity** parametric maps, respectively. Vorticity is a measure of rotation, whereas helicity relates to the knottedness in fluids, and they can be used for assessing the degree of flow non-laminarity.

**Vector Inspection**

The loaded velocity vectors can be inspected with the **QC** button. It shows a window with the projection of the flow direction vectors together with a color-coded direction image. Red indicates diffusion along the x-axis (left-right), green diffusion along the y-axis (posterior-anterior), and blue diffusion along the z-axis (inferior-superior).

Note that the time frame has to be selected before starting Vector QC, as the velocity vector field is four-dimensional.
Please proceed with the Tracking button in the lower right.

3.5 Calculating the Streamlines/Pathlines

When arriving at the TRACKING page, the image area shows a fusion of the Angio image (A) and the masked Angio image for highlighting areas with high flow. Also shown in the overlay are the VOIs.
Tracking Configuration

The area in the lower right serves for defining the tracking algorithm and its parameters. Two types of tracking options are available for selection:

1. **STREAMLINE**: is a curve that is instantaneously tangent to the velocity vector of the flow. It shows the direction in which a massless fluid element will travel at any time point.

2. **PATHLINE**: is a trajectory that an individual fluid particle follows. It can be thought of as recording the path of a fluid element over a certain period. The direction the path takes will be determined by the streamlines of the fluid at each moment in time.

Two algorithms are supported for any of the tracking option and can be selected in the Method list:

1. **EULER**: Tracking proceeds using a fixed step size along the local fiber orientation [2]. With nearest-neighbor interpolation, this method may be very similar to FACT, except that the step size is fixed, whereas FACT steps extend to the boundary of the next voxel (distance variable depending on the entry and exit points to the voxel).

2. **4th order Runge-Kutta** method [2]. The step size is fixed, however the eventual direction of the step is determined by taking and averaging a series of partial steps. Interpolation determines how the fiber direction is determined at a given point in space.

The Interpolation setting is not relevant for FACT. For EULER and RK4 the choices are

1. **NN**: Nearest-neighbor interpolation using the local voxel data directly.
2. **NC**: Probabilistic nearest-neighbor interpolation, similar to the method proposed by Behrens et al [3]. The data is not interpolated, but at each point one of the 8 voxels neighbors is randomly chosen. The probability of choosing a particular voxel is based on how close the point is to the center of that voxel.
3. **TRILINEAR**: Linear interpolation of the vector components using the 8 neighbors.

Further parameters and options determine how tracking proceeds and when it ends:
### Step size
Distance between points on the streamlines. Not used for FACT as it does not apply interpolation.

### Min velocity
Track ends if the velocity amplitude drops below this threshold.

### Min length
If checked, streamlines/pathlines shorter than the minimal length specified are discarded.

### Max curvature
If checked, streamlines/pathlines are terminated in case the curvature exceeds the specified angle within the length specified. If length = 0.0, the curvature is checked at each step.

### Colour by
The streamlines can be functionally colored as follows:
- **SEED**: Using the color of the SEED VOI.
- **END VOI**: Using the color of the END VOI.
- **DIRECTION**: Each segment of the track has a color combined from its direction cosines values: Red, Green, and Blue from direction cosines x, y and z respectively.
- **VELOCITY**: Using the velocity amplitude value along the track for coloring.
- **Last VOI**: using the color of the last VOIs in the list.

### Frames
Tracking is relatively time-consuming. With the box enabled it can optionally be restricted to a certain section of the acquisition.

### Masking
An additional means for restricting the generated streamlines/pathlines is by checking the Mask box and defining a mask. The elements in

- **Generate a mask from the ANGIO STATIC image by applying the specified threshold and include it in the image list B for viewing.**
- **Open the current mask in the VOI editor and convert it into a VOI. The VOI can be edited, and the mask finally updated from the VOI.**
- **Select an external mask file.**

**Note:** Please first perform tracking without masking, and only afterward gradually increase masking, if necessary.

After configuring all tracking parameters, please proceed with the blue action button. Based on the selected Tracking option, the streamlines or the pathlines are calculated and visualized on the 3D page.

### References

3.6 Visualizing the Streamlines/Pathlines in 3D

The 3D page initially shows the streamlines together with the VOIs.

Note the elements in the Scene tree in the upper right. An objects can be selected in the tree its appearance changed in the lower part. Please refer to the PMOD 3D Tool Users Guide for details about the rendering.

Initially the streamlines/pathlines of the first frame are shown, when typically there is low flow. To visualize the pattern at later times select the Streamlines/Pathlines entry in the object tree, activate its Plot properties panel, and drag the Frame slider or start a cine.

Image planes can be added by clicking at Planes in the tree, activating Add planes, opening the planes locator and finally selecting the MAG image. The plane location can easily be changed by clicking into the Select planes window, and the color on the Input panel.
3.7 4D Flow Report

The Report button on the 3D page composes a report as illustrated below:

The report is organized on six pages: **Values, Velocity, Vorticity, Helicity, Energy loss, Tracks statistics**

The **Values** page lists on the upper part the statistics of all defined VOIs. Shown is the maximum of all the VOI averages calculated along the time axis. The lower part shows a capture of the current 3D scene.
The **Velocity** page shows the plots of the averages **VELOCITY** amplitude for each time point in each VOIs:

The **Vorticity** page shows the plot of the average **VORTICITY** for each time point in each VOIs:

The **Helicity** page shows the plot of the average **HELICITY** for each time point in each VOIs.
The **Energy loss** page shows the total sum of the ENERGY LOSS for each time point in each VOIs.

The **Tracks statistics** page shows the average TRACK LENGTH in all SEED VOIs along time axis. The mean curvature (CRV[dgr]) is calculated and displayed for each Neutral VOI at each time point:
As usually, the report can be printed or captured in various formats via Print report. Save results serves for saving the numeric values, which can also be exported via Copy to Clipboard.

3.8 Creating VOIs along the Streamlines/Pathlines

It may be of interest to evaluate some of the images along the streamlines. The solution offered by PGEM is to convert the streamlines into VOIs, which can externally be used for statistics.

Please activate the Streamlines VOIs (or Pathlines VOI depending on the context) button on the 3D page to convert the streamlines into VOI outlines and open them in a VOI editor window.

Here they can be modified using the standard PMOD VOI tools, and saved for later usage.
3.9 Saving the Streamlines/Pathlines

The generated streamlines can be saved in vtk format for use in the standalone 3D tool, or for use with a third-party program. Please make sure that the 3D page is selected, as the saving functionality is page-specific. Then select the saving button in the taskbar to save the streamlines.

Correspondingly, already existing vtk track definitions can be loaded or appended as illustrated below.

3.10 Protocol File

The configuration of the 4D Flow analysis can be saved with the Save Protocol button in the status line. The adjacent Load Protocol button allows retrieving the configuration and repeating the analysis.
4 Generation of Dynamic PET Phantom Images

There is a particular workflow within PGEM for the generation of dynamic PET images representing defined kinetics in well-defined anatomical structures.

Input to this workflow is a set of VOIs which define the target anatomy. Additionally, the time course of the activity concentration (TAC) in each VOI must be provided. Such data can for instance easily be obtained by applying the PNEURO tool to an experimental dynamic PET study, optionally with an MRI.

Within PGEM, the VOIs are first transformed into an anatomical model. For each pixel within an anatomical structure the corresponding TAC is then used, optionally with some noise added, to generate a synthetic dynamic series with well-defined noise level. Finally, the images can be saved and, for instance, used to study the performance of a parametric mapping method in PXMOD.

4.1 External Preparations

Before starting the phantom generation process in PGEM two files should be prepared

1. a set of VOIs which define the anatomy,
2. a set of time-activity curves named exactly the same as the VOIs.

4.1.1 Phantom VOIs

The example below illustrates the results from analyzing a DASB PET/MR data set using the Brain Parcellation methodology in the PNEURO tool.

The VOIs can be saved with the button indicated by the upper arrow. The corresponding TACs are calculated with the Statistics button in the lower right.

4.1.2 Phantom Dynamic TACs

The resulting TACs are shown on the Statistics page of PNEURO and can be saved with the button indicated below.
4.2 Dynamic PET Image Creation

4.2.1 Anatomical Model Definition

The first step is to create a new model on the Models page. This can be done most easily by activating the + button in the taskbar and selecting the VOIs file followed by choosing Create new model in the dialog window as illustrated below.

A dialog window opens for defining the name and the timing of the series.

Timing from Existing Dynamic Image

The timing may be extracted from an actual image by selecting it with the open button and then activating Get from file. The image is read and displayed in a window.
After confirmation with Set the timing is opened for editing and confirmation with Ok.

Timing Specified per Dialog Window
The alternative way for timing definition is entering the **Number of frames** and opening the timing window illustrated below with the button:

When closing the model creation window with Ok, another window appears, requesting model settings. In this context it is easier to avoid loading the image, so it can be removed from Edit VOIs Image file A with the x button.
After closing with **Set and Load** the model is ready for the VOIs to be added. To do so, please confirm the dialog window with **Yes**.

**Note:** The timing specified should exactly correspond to the timing of the TACs.

### 4.2.2 VOI Addition

A dialog window comes up for defining VOI properties. This is useful for hierarchical anatomical models. For the generation of the dynamic PET phantom the **Surface estimation** should be switched off, while all other settings can remain as is.

After closing this dialog window with **Ok**, the VOIs are added and a confirmation message appears.
4.2.3 Anatomical Model

At this time the model is prepared and shown on the Models page. The left area provides an impression of the VOIs, the right area the incorporated structures in a tree arrangement. Note the indicated arrow button for unfolding/collapsing the tree, and the buttons for checking/unchecking all the structures.

It is important that all structures relevant for the phantom images are checked.

Please continue with the Phantoms button.

4.2.4 Phantom Creation

The Phantoms page lists the structures in the upper right. The timing is already defined, but could be changed in the lower left.
The next steps consists of loading the prepared TACs with the Load TACs button, such that they are shown in the Regional TACs curve panel.

Two options may be configured as illustrated below:

- The introduction of radioactive decay,
- The addition of noise.

The choices for configuring the noise properties are the same as the ones available in the PKIN tool for weighted fitting and Monte Carlo simulations. Once Add noise is checked, the random number distribution can be selected in the list. Consequently, a dialog window is opened for specifying the noise standard deviation. Please refer to the help entry for details as indicated below.

As the Calculated weighting takes the decay into account, the proper isotope has to be selected form the list. otherwise it has no impact. Note that the Scale factor scales the standard deviation, not the variance.
If the generated data should represent non-decay corrected data, the **Decay** box needs to be checked, and the appropriate isotope selected.

**Create Synthetic Study** starts data generation and shows the result in the lower right, where they can be saved using **Save SYNTHETIC**.
5 Geometric Model Construction and CFD Simulation

PGEM module is aimed to provide user with a comprehensive tool for fast and intuitive processing of 3D/4D image datasets from a wide range of scan modalities (MRI, CT, PET, microCT ...). The technical core of PGEM is converting 3D images in models.

This chapter describes the workflow for a geometric model construction, CFD simulation and visualization of the CFD results.

5.1 Geometric Model Construction

The geometric model is representing the functional core of the PGEM tool. It needs for input only images and consists of volumes of interest (VOIs) and auto-generated STereoLithography structures (STLs). Once the model is created and saved, it gives the user the possibility to focus only on applications. Still, the user can redefine the model structures at any time point. This "stop in the middle" of processing data, represented by the model, is a big advantage: the user does not have to repeat the path image data-data segmentation because everything is available within the model.

The model can be used to create:

- Phantom Simulation as explained in the dedicated chapter.
- Education-driven application: educational 3D visualization of structures complemented by graphics, pictures and text - knowledge base associated with geometry. It is similar to an atlas of human physiology and is important for education and research as well.
- Surface meshes suitable for CFD modeling. The surface meshes are compatible with CFD solvers like OpenFoam and Fluent. The user can create CFD cases directly through PMOD for OpenFoam. The results can be readily visualized in the interactive PMOD 3D environment in order to obtain new understanding about the blood-flow behavior. The visualization is based on volume rendering and aims to convert nonvisual data into readable and recognizable images.

PGEM is a promising solution which supplements the limited spatial and temporal resolution of imaging data with physically based fluid dynamic.

5.1.1 Workflow Organization

The geometric model construction starts on the Model page with the creation of a new empty model and the definition of the time and the input image.

The input image is loaded and displayed on the VOIs&Tracking/Image sub-page. If necessary, the image can be cropped before starting the VOI definition.

The work continues on the VOIs&Tracking/VOIs sub-page with the VOIs definition.

Finally, the VOIs (geometry) are added to the model in the model builder interface. This will result in the model tree structure. STL structures are auto-generated and saved for each VOI added to the model. Optionally, additional characteristics can be defined for each element present in the model tree structure.

5.1.2 Start of the Model Definition

The first step in the building process consists in creating a new empty model on the Models page. This can be easily achieved by activating the Create new model icon as illustrated below:
A dialog window opens and allows defining the name and the timing of the series. The name of the model is defined in the **Enter name** field: e.g. **BrainAneurysm**. Use the **Empty** radio button to create an empty model.

The **Duplicate existing** radio button allows making an exact copy of one of existing models in the list as illustrated below:

The duplicated model will be stored with the new name provided in the **Enter name** field. The modifications done in the duplicated model will not affect the source model.

**Continue with the time definition for the model.**

### 5.1.3 Time Definition

The models can be static or dynamic. The time definition for the **Empty** model is done in the bottom section of the window as illustrated below:
**Note:** The timing definition cannot be modified when **Duplicate existing** radio button is used to create a new model. In such case the timing defined in the source model will be set for the new model.

### Timing Specified per Dialog Window

The specification of timing is done entering the **Number of frames** and opening the timing window illustrated above with the **button.**

The **Save Time/Retrieve Time** buttons allow saving/retrieving the timing of the dynamic frames to/from a file. It is possible to select a subset of frames in the **Frame selection** specifying the frames number in the **from** and the **to** text fields. The specified frame **duration** in samples is **Set to Frame selection** or **Set to all Frames.** The **Trim durations button** ensures that the end times are not after the following start times. The **Remove gaps & overlaps** button ensures that there are no empty time intervals between frames and that the start or end times do not extend beyond the previous or following times respectively.

A number of 1 frame is set to the **BrainAneurysm** model. It indicates a static model.

### Timing from Existing Dynamic Image

The timing may be extracted from an actual dynamic image by selecting it with the open button and then activating **Get from file.** The image is read and displayed in a window.
After confirmation with **Set** the timing is opened for editing and confirmation with **Ok**.

**Note:** The timing definition cannot be edited later on.

Close the **Create Model** window with **OK** to continue with the image definition.

### 5.1.4 Image Definition

The following steps are described using the **Aneurysm 1** series of the **PGEM3** subject available in the Demo database.

**Image Definition in Model Settings Window**
When closing the CREATE Model window with Ok, another window appears, requesting image(s) definition:

The Edit mode tab allows defining two images: image A and image B.

Image A is the image that will be loaded when the Load & edit button is activated. It will be considered the basic image for editing VOIs. Image B, if defined, has to be matched to the image A beforehand, as the two will be shown using the fusion mode on the Images page.

With the Reorient to Anatomical Position box enabled, the images are brought into the radiological HFS orientation after loading.

Use the button to add a VOI file to the model at this stage. Please note that the VOIs have to be previously outline and saved as .voi file on image A or B.

The Reference Image tab provides additional settings:

It allows defining a reference image which can be A or B or any other image in the same space. The Image file name field allows assigning a name to the image. The default name is assigned based on the model name followed by the suffix _image_1 as shown in the example above. The preview button allows visualizing the images set for the model. Enable the Reorient to Anatomical Position box to bring the image into the radiological HFS orientation. Activate the Copy to model as Reference Image (Nifti) green button to copy the image in the model folder as a Niftii image. A confirmation dialog will appear asking the confirmation of the saving procedure. Close the dialog with YES to save the reference image in the model folder.

In the example above the same image was defined as image A and as Reference Image.

Activate the Load & edit button to proceed to the VOI definition step:

The program switches to the VOIs & Tracking page, Image sub-page while loading and displaying the image defined as A:
The images data with large field-of-view can be reduced in order to save RAM and optimize processing. This can be achieved enabling the Crop box. A yellow crop volume appears on the image. Place the yellow crop volume by clicking in the center of the area of interest such that it is fully enclosed. The edge Size in [mm] can be adjusted for each direction by selecting the size in the corresponding list. The Crop button initiates cropping, whereby the original data are replaced. If cropping is not initiated manually, a request will be shown when proceeding to the VOIs step.

**Note:**

To use the Reference image for the VOIs definition select the corresponding entry under the Load & edit selection list, as illustrated below:

Image Definition by Model Image Loader

When no image is defined in the Model Settings interface, after pressing Load & edit button, the user will be asked to provide such configuration. In case there is no intention to provide such configuration, the user has to manually select the VOIs & Tracking page Image sub-page, enable the Model radio button and load an image configuring the available Load button. This image will be used for the VOIs definition in the next step.

Continue the VOI definition activating the VOIs action button.
5.1.5 VOI Definition

When arriving at the VOIs sub-page a fusion image is shown to the left which is composed of the image A and image B. In the current example only image A was defined in the Model settings window.

The VOIs are created with the usual PMOD VOI tools. Please refer to the PMOD Base Functionality Guide for information related to volume-of-interest (VOI) definition. In the example illustrated below the Automatic Isocontouring Detection tool was used for outlining the first VOI:

Please note that as soon as a VOI is available in the VOIs list the Add Group to Model green button becomes active.

**Note:**

There is no limitation to the number of VOIs to be defined. It is recommended to define the VOI on the plane which is most perpendicular to the vessel axis.

For CFD simulation it is recommended to create individual contour ROI: e.g. each branch of a bifurcation should be a separate VOI. However, different VOIs can overlap. In case of joining structures (e.g. bifurcation) the VOIs should overlap on a couple of slices.
5.1.6 VOI Addition

Start adding VOIs to the model selecting the **Group** tab as illustrated below:

Activating the **Add Group to Model** green button the currently selected VOI is added to the model. Please note that more than one VOI at a time can be added to the model. A confirmation window appears and allows choosing to which model the geometric structures should be added:

Activate the **Yes** button to confirm the default settings which point to the currently selected model on the **Model** page. The model builder interface comes up and allows defining the VOI destination and the properties in the anatomical model.
The appearance of the model builder interface is adjusted according to the Current VOI setting. The window is organized in three main sections:

1. **Selected VOIs destination**: allows defining the Anatomic Group and the Structure.
2. **Surface estimation**: allows defining settings to build the STL structure of the model.

**Selected VOIs Destination**
The Selected VOIs destination section requires definitions for the Anatomic Group and Structure. When VOIs are added to the model for the first time, these settings are <Empty> and New names need to be provided in the dedicated field. To this end, the Brain Vessels and the Aneurysm were set as Anatomic Group and Structure respectively.
The settings available in this section are summarized in the table below:

<table>
<thead>
<tr>
<th>Anatomic Group</th>
<th>Corresponds to the highest hierarchy level in the model tree structure: the Group</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Allows defining the model Anatomic Group to which the VOIs will belong. To define a new Anatomic group the New checkbox has to be enabled and the name specified in the dedicated text field. Confirm and add the entry to the list with the Enter button on the keyboard.</td>
</tr>
<tr>
<td></td>
<td>The drop down arrow becomes active when more than one anatomic structure was defined. It allows selecting among the currently available Anatomic Group.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Structure</th>
<th>Corresponds to the second hierarchy level entry in the model tree: the Structure.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Allows defining the model Structure to which the VOI will belong. To define a new Structure the New checkbox has to be enabled and the name specified in the dedicated text field. Confirm and add the entry to the list with the Enter button on the keyboard.</td>
</tr>
<tr>
<td></td>
<td>The drop down arrow becomes active when more than one structure was defined. It allows selecting among the currently available Structure.</td>
</tr>
</tbody>
</table>

**Note:** Multiple Structures can belong to the same Anatomic Group. To this end turn off the New checkbox for the Anatomic group, select one of the Anatomic Group available on the list and proceed with the definition of a new Structure as described above.
It is possible to edit the VOI color and the appearance of the VOI in the model timeline while adding VOIs to the model.

<table>
<thead>
<tr>
<th><strong>Place VOI in the model's timeline</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Allows changing or editing the color of the VOI(s) to be added to the model structure.</td>
</tr>
<tr>
<td>Allows setting the model frame where the VOIs will appear. This setting is supported by the From and To fields.</td>
</tr>
<tr>
<td>Please note that the frame number for the model has to be defined previously.</td>
</tr>
</tbody>
</table>

**Surface Estimation**

The Surface estimation section allows defining characteristics for the STL model structures. While adding VOIs to the model surfaces (STL files) are generated automatically with the Surface estimation box enabled. These STL files will be stored in the model folder and will be used for 3D model presentation. Generating them will allow rendering faster the whole model in 3D.

**Note:**

Please note that these STL files are not the STL files used as surface meshes while creating CFD case process.

With Smooth VOIs radio button active the settings are as follows:

<table>
<thead>
<tr>
<th><strong>Min pixel size of VOI to add</strong></th>
<th>Displays the pixel size [mm] in the data</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Smoothing (FWHM)</strong></td>
<td>It is a Gaussian filter and allows smoothing the data before creating the STL structure. The default value corresponds to the pixel size in the data. It is recommended to use a smoothing value 3 times the minimum pixel size.</td>
</tr>
<tr>
<td><strong>Note:</strong> Please note that for complex VOI shape less smoothing should be applied</td>
<td></td>
</tr>
<tr>
<td><strong>Segmentation Threshold</strong></td>
<td>All pixels with the value above the threshold are included in the segment. The default value is 0.35. When the Optimal check box is enabled the threshold will be calculated automatically such that the volume of selected VOIs is preserved.</td>
</tr>
<tr>
<td><strong>Reduction</strong></td>
<td>If enabled, allows creating simplified mesh based on data simplification. The defaults value is 2 and is representing the smallest value to be defined.</td>
</tr>
<tr>
<td>Initially, the program will use this number and multiply the pixel size of the data with this value. Finally, the mesh will be generated on the new data. It is a faster procedure and the results will consist in mesh being less precise. Please note that if only this check box is enabled the generated files will be bigger and change in topography of the mesh will occur.</td>
<td></td>
</tr>
<tr>
<td><strong>Decimation</strong></td>
<td>Allows simplifying the mesh by removing triangles from the surface mesh without changing the mesh. Please note that in such case the simplification procedure is done after the mesh is created. The defined value should be between 0 and 1. The number represents percentage of the triangles to be thrown out. Usually „0.5“ is a good compromise. The smaller the number of triangles, the faster is the 3D visualization.</td>
</tr>
<tr>
<td>It is a slow procedure which generates smaller files. Decimation only changes the number of STL nodes,</td>
<td></td>
</tr>
</tbody>
</table>
particularly the vertices of the surface mesh. However, no change in the mesh geometry occurs, preserving thus the mesh topology.

In alternative, the **Use segmentation iso-surface level** radio button can be used. It is recommended to use this option when the VOI was outlined using the isocontouring VOI tool and the image data loaded is the image data on which the VOI was created. With these two conditions fulfilled, the threshold value used for VOI isocontouring has to be set as **Original Segmentation Threshold**.

**Note:**

The iso-surface is recommended to be used for big geometries like skull.

In the current example the **Smooth VOIs** radio button is enabled. The **Smoothing (FWHM)** parameter is set to 3 times the pixel size (1.2 mm), **Decimation** is set to 0.5 while **reduction** is avoid.

Activate the **OK** button to start adding the VOI available in the **Current VOI** list to the model. The VOI will correspond to the lowest hierarchy level in the model tree structure: the **Element.** STL structures are auto-generated and saved for each VOI added to the model. This is encoded in the model tree by the letter **s** between round brackets which precede the **Element** and the **Structure** name. The result of the above processing is illustrated below:

5.1.7 **Anatomical Model**

The anatomical model consists of VOIs (**Elements**) and auto-generated STereolithography structures (**STLs**). The process of adding VOIs to the model results in the model tree structure. Multiple models may coexist in the same time but only one model is visible at a time.

The **Models** page has the layout illustrated below:
The model is available for selection in the model list, in the upper right. The model tree structure is displayed in the right panel, under the model selection list.

The viewport area to the left shows the selected elements in the model tree, whereas the characteristics and the properties of the visualized objects can be previewed and/or edited using the panel to the right.

5.1.7.1 Tree Organization of the Anatomical Model

The result of adding VOIs to the model is structurally organized in the model tree on the Models page.

The selection of a subset of elements in the model is supported by the dedicated panel as illustrated below:

The model structure is organized as a hierarchical tree on three levels: top, middle and low level. The hierarchy corresponds to the Selected VOIs destination settings defined in the model builder interface during the VOI addition.

The branch on the top level is represented by the Group, e.g. the BrainVessels, as indicated in the left panel. It corresponds to the Anatomic Group defined in the model builder interface.

The full tree can be opened easily by the button indicated in the right panel. The middle level entry in the model tree is represented by the Structure node, e.g. the Aneurysm. It corresponds to the Structure defined in the model builder interface.
The single entry in the tree is called *Element* and corresponds to a VOI in the VOIs list. It represents the low level entry in the model hierarchy, e.g. *Vessels*.

In general, the *Structure* node consists of at least one *Element*. The *Group* consists of at least one structure node. A model may consist of one or several *groups of structures*. An example of such model (*HEART*) is illustrated below:

![HEART model illustration](image)

It consists of three groups: *Arteries*, *Veins* and *Muscles*. Each group contains multiple structure nodes: e.g. the *Left heart* and *Right heart* in the *Muscles* group.

Round brackets are used to add encoded information for each entry in the model tree. Such information precedes the name of the entry and is summarized in the table below:

<table>
<thead>
<tr>
<th>s</th>
<th>Stands for <em>surface mesh</em> and indicates that a VOI was added to the model as geometry. STL structure is auto-generated and saved for each such VOI. Note that each middle and low level entry in the model tree has such information available. When an element is selected in the model tree a preview of the simplified mesh structure is shown in the viewport.</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>Stands for <em>case</em> and indicates that a CFD simulation result was linked to the selected entry in the model through the local cases manager. When such encoding is available for a selected entry in the model tree the button in the taskbar becomes active. It allows viewing the linked CFD case in the 3D interface like a polymesh (volume mesh):</td>
</tr>
</tbody>
</table>
Tree Manipulations

Branches in the tree can be opened/closed with the little arrows left to the branch names.

Tree Selections

The structures of interest are the ones with checked boxes to the left of their color code. Simply click into a box for changing the selection. In the example below the RV epicardium and RV endocardium have been de-selected.
If the selection of a branch is removed, all elements belonging to it will be de-selected.

5.1.7.2 Components of the Model Folder in PMOD

The model is stored in the **Structural model Path** defined in the PGEM configuration settings. The model directory contains two folders (**images** and **structures**) and two files as illustrated below:
1. The **image** folder contains the image copied to the model folder as the reference image in niftii format.

![Image Folder](image_folder.png)

2. The **structures** folder contains all the information related to the entries in the model tree.

3. The **atlasdir_*** file stores the frame time definition, reference image file information, settings used in the model builder interface.

4. The **model** file stores the information related to the PMOD version used to create the model and the definition of the model images.

**The Structures Folder**

The information related to each top level entry in the model hierarchy is organized in subdirectories with the same name. Their location is in the **structures** folder of the model directory as illustrated below:

![Structures Folder](structures_folder.png)

The number of the subdirectories present in the **structures** folder equals the number of top level entries in the model tree. In the example above a single subdirectory is present: the **brainvessels**. It indicates the model has a single top level entry.

The content of such subdirectory is shown below:
It contains:

1. The .voi file(s): the number of these files equals the number of middle level entries belonging to the same top level entry in the model tree. The name of the middle level entry in the model tree is used as the name of the .voi file: e.g. Aneurysm.voi.

2. The .XML file: the name of the top level entry in the model tree is used as the name of the .xml file: e.g. Brain_Vessels. It stores information of the edition dates and the name of the top level entry in the model hierarchy as illustrated below:

   ```xml
   <?xml version="1.0"?>
   <!--Put all new tags under <entry> level.-->
   <!--Order of the groups under the <entry> level, such as f.ex. <Name>, <Description> does not matter.-->  
   <!--Do not remove or rename default groups, which are: Name, Geometrical, Physical, Description and Images.-->
   <!--XML preserves the formatting of text between tags (it does not ignore spaces and empty lines).-->
   <!--Information between <description> tags must be put in between in such way that the text is not in the same line as opening and closing tags-->
   <entry>
   <Edition_dates>
   <First_addition_date>2021.02.24</First_addition_date>
   </Edition_dates>
   <Name>
   <model>BrainVessels</model>
   </Name>
   </entry>
   ```

3. The stl folder: contains the STL file of each middle level entry belonging to the same top level entry in the model tree. Particularly, for each .voi file the corresponding STL file is saved.

   Please note that a suffix, e.g. _f1 is appended to the name: it corresponds to the Place VOI in model's time line definition in the model builder interface. This indicates that the Place VOI in model's time line fields From and To was set to 1. The static models (1 frame model) will always have this setting set to 1. The STL files are used for 3D simplified preview of the model.

4. The characteristics folder: stores the definitions provided as characteristics for each entry belonging to the respective top level entry in the model tree.
Such definitions are stored separately for each entry level in .xml files. The number of .xml files equals the number of middle and low level entries belonging to that specific top level branch.

The name of the top level entry in the model followed by the middle level entry one is used as the corresponding name for the Structure .xml file: e.g. BrainVessels_Aneurysm.

The name of the top level entry in the model followed by the middle and low level entry ones is used as the corresponding name for the Element .xml file: e.g. BrainVessels_Aneurysm_Vessels.

In the example below is illustrated the content of the BrainVessels_Aneurysm_Vessels:

```xml
<?xml version="1.0"?>
<entry>
  <Edition_dates>
    <First_addition_date>2021.02.24</First_addition_date>
  </Edition_dates>
  <Name>BrainVessels_Vessels</Name>
  <Images>
    <Image>CircleOfWillis.jpg</Image>
    <Image>Aneurysm.jpg</Image>
    <Image>CircleOfWillis.jpg</Image>
    <Image>Aneurysm.jpg</Image>
  </Images>
</entry>
```

Each .xml file stores information about the edition dates, the level entry name encoded as described above, the text description and the images file names. The images, if defined, are stored as .jpg files with the specified name: e.g. CircleOFWillis and Aneurysm in the example above.

**Note:** To share a model with another PMOD user simply copy the model folder (e.g BrainAneurysm) to another PMOD installation in the Structural model Path defined in the PGEM configuration settings.

### 5.1.7.3 Definition of the Characteristics for the Model’s Elements

Use the characteristics section on the Models page to provide images and text description for the entries in the model tree. The characteristics of the models and the 3D visualization are representing the education-driven application of the models.

Select an entry in the model tree (1) and activate the View Characteristics button (2) to start the characteristics definition. A dialog window opens and allows allowing editing and adding captured images:
The window is organized in two main tabs, **Images** and **Description**. The characteristics can be easily edited using the buttons available in the bottom section of the window.

Please note the tooltips which provide short explanation of the buttons functionality.

Initially, only the description tab is active as no image is defined in the characteristics. Activate the **T** icon (3) to start **Edit description**. Please note that text sections entered in this area should be formatted using HTML tags (4): `<b>` and `</b>`. In the example above two sections are formatted using the HTML tags:

* `<b>Circle of Willis</b>`

* `<b>Circle of Willis Structure</b>`

Close the dialog window with the **OK** button (5) to transfer the content of the edit dialog to the **Characteristics**. Use **Cancel** button if no text should be transfer to the **Description** tab.

Activate the **+** button to start adding an image. Its activation opens a dialog window as illustrated below:

It allows defining the **Name** for the image. Once the image is selected the activation of the **Add** green button stores the image as a characteristic. Select the **<** button to preview the image added as characteristics. Finally **Close** the window. The image is available on the **Images** tab and the assigned name is displayed on the status line of the window as shown below:
The arrow button becomes active when multiple images are added as characteristic to the selected element. It allows selecting the image to be displayed on the Images tab. Activate the Delete image button to discard an image.

Use the Close button to exit the Characteristics window.

**Note:** The recommended graphic size for images is 600x600 pixels.

### 5.1.7.4 Definition of the Properties for the Model’s Elements

The properties section on the Models page allows modifying the properties of the element selected in the model tree:

The Rename text field allows changing the name of the entry in the tree by simply typing a new name. To set the new name it is mandatory to press the Enter key on the keyboard. A dialog window opens: activate the OK button to confirm the change.

To eliminate an entry in the model tree, select the element or the node and activate the Delete button. A dialog window appears allowing to Completely remove from the model the selected entry or to Delete the definition from the specified frame.

The Property available for selection is as follows:

1. VU is representing the user defined units.
2. YM stands for Young's module.
3. AI stands for acoustic impedance.

To assign a certain e.g. VU value to one or more entry in the model tree please proceed as follows:
1. Select the element in the model tree. For multiple selection use Shift+click or CTRL+click.

2. In the text field aside the Property selection (e.g. VU) type the value (e.g. 100).

3. Confirm the action with the Enter button from the keyboard.

**Note:**

In case property value is entered for a top level entry it will affect all the nodes beneath it.

The property will be applied to all selected nodes: the selection is referring to the node, not to the checkbox aside the node.

To define different properties to the elements in a node start the definition with the highest hierarchical node and then continue with the lower level nodes beneath it.

If no value is entered for the current selection the property field displays "0".

Below the properties panel additional functionality buttons are available:

<table>
<thead>
<tr>
<th>Button</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Icon" /></td>
<td>Hide/show the properties panel to free some space in the user interface. With the panel hidden, the icon changes to <img src="image2" alt="Icon" />. When this button is activated, the panel is shown again.</td>
</tr>
<tr>
<td><img src="image3" alt="Icon" /></td>
<td>Allows starting the movie when dynamic models are defined. The text box aside allows defining the movie delay. The slider <img src="image4" alt="Icon" /> allows selecting for preview a specific frame in a dynamic model.</td>
</tr>
<tr>
<td><img src="image5" alt="Icon" /></td>
<td>Allows stopping the movie</td>
</tr>
<tr>
<td><img src="image6" alt="Icon" /></td>
<td>If enabled shows 3D Cartesian coordinate system.</td>
</tr>
<tr>
<td><img src="image7" alt="Icon" /></td>
<td>Allows selecting the number of the 3D vertices to be displayed in the view port. The higher the number the less vertices will be shown.</td>
</tr>
<tr>
<td><img src="image8" alt="Icon" /></td>
<td>List selection for changing the background color.</td>
</tr>
<tr>
<td><img src="image9" alt="Icon" /></td>
<td>Allows rotating 360 degrees the 3D simplified model preview in the view port.</td>
</tr>
</tbody>
</table>

5.1.7.5 Interactive Exploration of the Model

Start on the Model page by selecting a model and then the elements in the model tree to be visualized in 3D.

In the example illustrated below the Muscles/Left heart branch in the HEART model tree is selected for exploration:
Activate the 3D button to start the interactive exploration of the model.

The program will render the selected elements of the model and show them in the 3D page as: **STL, Volume Mesh or Volume Mesh +STL complement.**

The **STL** rendering result is illustrated below:
Note that default properties of the elements in the 3D View tree were changed for exploration purposes. Please refer to the PMOD 3D Rendering Tool Users Guide for rendering-related information.

5.2 CFD Simulation and Implementation in PGEM

The CFD simulation is started activating the Create CFD case green button.
A dialog window opens and allows selecting the starting point to create the CFD case: a geometric model, an existing surface mesh (SM), an existing volume mesh (VM) or loading a previously saved case.

With the Load image data box enabled it is possible to Set paths for velocity and magnitude image data. The images will be used in the 3D interface for the generation and visualization of the profiles on the cutting planes.

Start CFD case from the Model
This is the most detailed workflow and is based on the selected geometry in the model. The other workflows are essentially simplified variants of it but require the availability of the starting point files usually created in a detailed workflow.

The creation of the CFD case requires performing the following steps in order:
1. select the geometric element in the model
2. create surface meshes based on the selected model geometry
3. create volume meshes based on the surface meshes
4. set boundary conditions for the simulation problem
5. set parameters for the solver (e.g. OpenFoam)

It is advised saving all the steps definitions in a case file once the CFD case creation is completed. This allows retrieving the case configuration at any later time point. The case definitions can be modified to try variations of the processing parameters.

Start CFD case from existing Surface Mesh (SM)
If a SM is available with properly define solids the following steps are required:
1. load the SM file.
2. create volume meshes based on the surface meshes
3. set boundary conditions for the simulation problem
4. set parameters for the solver (e.g. OpenFoam)

Start CFD case from existing Volume Mesh (VM)
If a VM is available on the file system or in the local case manager, the following analysis steps are required:
1. load the VM file.
2. set boundary conditions for the simulation problem
3. set parameters for the solver (e.g. OpenFoam)

Start CFD case from Existing Case
To start the CFD simulation from an existing case the *.cfdCase file has to be loaded and send to the solver. Such file includes surface meshes definition, the volume mesh settings as well as the definitions of the boundary condition and the parameters for the solver.

4D Flow CFD case
The creation of a CFD case based on 4D Flow data requires performing initially the 4D Flow analysis up to the VOI definition step. Next, the user has to switch to the Model page and select the anatomical model built using the same 4D Flow images data. The CFD case creation may start using any of the above scenarios: from the model, from an existing SM, from an existing VM or from an existing case.

In the 4D Flow CFD case study the velocity profile can be calculated and visualized for the defined cutting planes.
The **4D FLOW CFD** model will be used for the description of all CFD case creation.

### 5.2.1 Create CFD Case from the Geometric Model

#### 5.2.1.1 Geometry Selection in the Model

Start the workflow selecting the elements in the model tree. In the example illustrated below all the elements in the **CFD simulation VOIs** branch are selected in the **4D Flow CFD** model tree:

The creation of the CFD case is started with the **Create CFD case** green action button. A dialog window opens allowing to select the starting point for the case: enable the **Start from the model** radio button. The appropriate **Create CFD case** interface appears. It provides a step-by-step guidance through the CFD setting process:
The definition steps are mandatory and have to be completed for the CFD process.

**Note:**

The geometric model can be used to repeat simulation as VOIs editing step is a fast process. Starting from the base geometry a series of different geometries can be built: narrow, widen, with extra branches. The new geometries (VOIs) can be easily added to the model, as explain [previously](#), and use to run CFD experiments.

### 5.2.1.2 Surface Mesh Creation

The first step in the CFD case preparation is the creation of the surface meshes. Initially, only the **Surface Mesh** tab is active as illustrated below:
The SM settings are described in the table below:

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Min pixel size of selected structure</strong></td>
<td>Displays the pixel size [mm] in the data.</td>
</tr>
<tr>
<td><strong>Smoothing (FWHM)</strong></td>
<td>It is representing a Gaussian filter and allows smoothing the data before creating the STL structure. The default value corresponds to the pixel size in the data. It is recommended to use a smoothing value 3 times the minimum pixel size.</td>
</tr>
<tr>
<td><strong>Segmentation Threshold</strong></td>
<td>All pixels with the value above the threshold are included in the segment. The default value is 0.35. When the <strong>Optimal</strong> checkbox is enabled the threshold will be calculated automatically such that the volume of selected elements is preserved.</td>
</tr>
<tr>
<td><strong>Decimation</strong></td>
<td>Allows simplifying mesh by removing triangles from the surface mesh without changing the mesh. Please note that in this case the simplification procedure is done after the mesh is created. The value ranges between 0 and 1. The number represents the percentage of triangles to be thrown out. Usually a value of 0.5 is a good compromise. The smaller the number of triangles, the faster will be the 3D visualization. It is a slow procedure which generates smaller files. <strong>Decimation</strong> only changes the number of STL nodes, particularly the vertices of the surface mesh. However, no change in the mesh geometry occurs, preserving thus the mesh topology.</td>
</tr>
<tr>
<td><strong>Save after generation</strong></td>
<td>When enabled allows saving the generated STL file in the <strong>surface-meshes</strong> folder. This folder is located in the directory</td>
</tr>
</tbody>
</table>
set as **Case path (local)** in the PGEM configuration settings.

The SM STL files stores information related with the Bounding Box and Point in mesh location. This information is used for the calculation of the VM.

<table>
<thead>
<tr>
<th>Define boundary condition plane</th>
<th>When enabled allows starting the definition of the cutting planes. The cutting planes definition is mandatory for the CFD simulation. When disabled allows creating SM for visualization purposes. Such SM are used to create VM of selected geometries for visual presentations and cannot be used for CFD simulation.</th>
</tr>
</thead>
</table>

In the example above the **Smoothing** was set to 3 mm while the default settings were used for **decimation** 0.5 and **segmentation threshold** 0.35. Toggle the **Save** and the **Display** checkboxes in order to save and preview respectively the result of the definition of the cutting planes.

Enable the **Define boundary condition plane** checkbox to start the definition of the cutting planes.

PMOD supports the 3D method for the definition of the cuttings planes. Activate the **Generate surface mesh** green button to start the generation of the cutting planes.

### 5.2.1.2.1 Cutting Planes Definition with 3D

It is recommended to use the oblique planes to create cutting planes for the CFD simulation. The oblique planes allow creating cutting surfaces which are perpendicular to the vessel axis.

The oblique plane needs to be interactively placed perpendicular to the vessel axis.

The **3D** interface for the definition of the cutting planes is illustrated below:
In the example above one oblique plane was used to create the cutting planes on the selected surface render object: the walls. The results are two entries in the View tree, in the walls branch: the Cutting_caps_1 and Cutting_caps_2.

For each cutting plane 1 and 2 a cap was created and named as inlet and outlet respectively.

When velocity and magnitude data are loaded when starting the CFD Case the Generate data profile button is active and allows generating the velocity profile for the selected cutting plane.

Activate the Generate data profile button to visualize profiles for the selected cap (e.g. inlet):

The Preview tab allows selecting one of the images available on the list. The profile for the Velocity Amplitude only can be exported in PMOD or Fluent compatible format activating the Export velocity profile button. With the push pin button in "append" mode the currently selected frame will be available for selection in the boundary condition settings step for the velocity (U) under the selection "data profile(image based)".
The **Mag** tab allows visualizing the flow magnitude:

![Image of Mag tab](image1.png)

The **Velocity** tab allows visualizing the magnitude (MAG), the Velocity AMPLITUDE, and separately the velocity in X and Y directions.

![Image of Velocity tab](image2.png)

To exit the **Data Profile for cap:[]** interface activate the **Close** button.

Close the 3D interface with the **OK** button to generate the surface mesh. The calculation process may take a couple of minutes.

Activate the **View surface mesh** button to visualize the SM in a 3D window as illustrated below:
The 3D interface allows exploring interactively the generated surface mesh. **Close** the window and continue with the VM creation. The cutting planes will be used to define the boundary conditions for the CFD simulation.

Please refer to the *PMOD 3D Rendering Tool Users Guide* for any further details about the oblique planes and cutting-related information.

### 5.2.1.3 Volume Mesh Creation

The **Volume mesh** tab becomes active once the SM is generated. In the process of SM creation the **Bounding box** and **Point in mesh location** parameters are calculated automatically and displayed as illustrated below:
It is mandatory that the **Bounding box** encloses the whole surface. The **Point in mesh location** represents coordinates of any point inside the SM. It is recommended to preserve the automatically calculated values for the **Bounding box** and **Point in mesh location** and optionally change the **Cell size** parameter. This parameter determines the size of the initial cube cells. In the example above the values calculated automatically are used for the simulation purposes.

**Note:**

The bounding box and location in mesh can be edited via image data - using the screwdrivers buttons.

The **Surface refinement** is representing the parameter that will steer division of cells of a volume mesh in the surface's neighbourhood. Particularly, the maximum number of divisions (level) will be applied to the curved parts of the surface while the minimum number of division anywhere else in the neighbourhood. It can be defined for each surface (walls, inlet or outlet) separately or on all surfaces selecting all boundaries. Please note that the parameters for a particular surface (e.g. walls, inlet, outlet) will overwrite the all regions settings. In this particular case the surface refinement was set for all boundaries to a max level of 3 and a min level of 2.

The **Volume refinement** is representing the number of cell divisions inside the whole volume of the geometry (each cell of the volume mesh). A level of 2 is set in the example above.

**Note:**

The default settings always build rather simple volume mesh and are recommended to be to use for the first estimation of a solution. The cells number of a volume mesh and its quality is always a compromise between accuracy of the solution and the time of calculation. Using a bigger number of cells does not change the quality of the solution.

With the **Finish processing at this stage** box enabled the **Generate volume mesh** green button becomes available. It allows calculating the volume mesh but exits the **Create CFD case** dialog. The result is an OpenFoam case storing only the volume mesh. To create a volume mesh compatible with Fluent solver activate the **Export to Fluent format** check box.
To continue with the CFD case creation select the boundary condition tab.

5.2.1.4 Boundary Condition Settings

In this step the boundary conditions are defined for the cutting planes and for the wall. There are two physical variables which characterize the fluid flow: the pressure and the velocity.

The Boundary Conditions interface is illustrated below:

The **Internal field** allows setting initial values for both physical variables. These values will be assigned to every cell in the volume mesh. It allows starting the simulation from field with non-zero values. In the example above the zero value is used both for the pressure and velocity in the CFD simulation.

The **P (pressure)** is a scalar and is defined as a kinematic pressure (pressure/fluid density) and measured in [m²/s²]. There are two types of pressure boundary conditions: **fixed value** and **zero gradient**. With the **fixed value** selection the value of the variable will stay as defined throughout the simulation. With the **zero gradient** selection it is assumed that there is no value gradient along the flow direction. Particularly, it is assumed that the flow is fully developed at this boundary, and the value will be calculated by the solver.

In the CFD simulation example above the **zero gradient** is set for the **walls** and the **inlet_plane** surfaces and a **fixed value** of 0 for the **outlet_plane** surface.

The **U(velocity)** is a vector and is measured in [m/s]. There are three types of velocity boundary conditions: **fixed value**, **surface normal fixed value** and **zero gradient**.

When the **fixed value** is set the value of the variable will stay as defined throughout the simulation. A similar situation is obtained with the **surface normal fixed value** selection but in this case the vector alignment is always normal to the surface for which the boundary condition is applied. The inflow is defined by a negative value. The outflow is defined as a positive value. The **zero gradient** significance is the same as for the pressure variable: assumes the flow is fully developed at this boundary, and the value will be calculated by the solver.

In the example above, the following settings are done for the **U** variable:
- **walls surface**: fixed value 0 0 0 (the so called no slip boundary condition)
- **inlet surface**: surface normal fixed value with a value of -0.26
- **outlet surface**: zero gradient

**Note:**

It is recommended NOT to set fixed constraints on the flow at one surface for both variables, e.g. pressure and velocity set as fixed values at the inlet.

When velocity profiles were generated and set for a selected frame for a cutting plane in the 3D interface (e.g. inlet), these profiles are available for selection as data profile (image based) boundary condition for the velocity. The View data profile button allows visualizing these profiles as illustrated in the capture below.
5.2.1.5 Solution Control

The last step consists in setting the parameters for the solver as illustrated in the capture below:

The number of Iterations is recommended to always be rounded to full tens. Toggle the write final result only radio button to save only the final solution for the simulation. Consequently, the wall share stress (WSS) will only be calculated for the last point when the dedicated box is enabled.

The best way for reproducing a simulation is to save the entire case configuration with the Save button in the lower left. All the information will be saved in a .cfdCase file.

The Case name field allows assigning a meaningful name to the CFD case. This facilitates the management of the OpenFoam remote and local cases. In the example above the AortaCase was assigned as the name for the CFD experiment.

With the Convert result to FLUENT data format box enabled the case will become compatible with the FLUENT software.

Activate the Execute case button. A confirmation panel appears listing the Case summary:

With the Download case to the local cases folder box enabled the CFD simulation results are automatically downloaded when calculations are completed. The Delete case.., box enabled allows deleting the case from the remote location.
Close the confirmation window with OK to send the CFD case to the OpenFoam interface and exit the CFD settings interface. A message will confirm the success of sending the case to the server or not.

5.2.2 Create CFD Case from a Surface Mesh

Start the CFD simulation activating the Create CFD case green button. A dialog window appears as illustrated below:

Toggle the Load existing surface mesh radio button to start the creation of the CFD case from a SM. Use the open button to load a SM file. To define the SM units select one of the list entries available for the Mesh is defined in. Optionally image data can be define to use for the visualization of the 4D Flow profiles. With the Load image data box enabled it is possible to Set paths for velocity and magnitude image data. To confirm the selected images activate the SET button.

Close the dialog window with the OK button to continue the creation of the CFD case. The CFD interface appears displaying the Volume Mesh as the first step of the analysis.
The **View surface mesh** button allows exploring interactively the loaded surface mesh in the 3D interface.

The **Bounding box** and the **Point in mesh location** parameters are initially set to 0 and 1 respectively. It is mandatory for the user to know these parameters and interactively type them in the dedicated text boxes.

The **Surface refinement** parameters are set to **all boundaries** to a **max** level of 3 and a **min** level of 2. This indicates that a **max**imum number of 3 divisions will be applied to the curved parts of the surface while a **min**imum number of 2 divisions anywhere else in the neighbourhood.

The **Volume refinement** is set to a **level** of 2. This indicates that each cell in the volume mesh will be divided in two.

Continue with the definition of the **Boundary Conditions** as explained previously.

Finally, define the parameters for the solver. Set the number of **iterations** to a full tens number, e.g. 100 and enable the **write the final result only** radio button to save only the final solution for the simulation. Enable **Calculate wall shear stress** box for WSS calculation.

**Note:** Assign a meaningful name to the CFD case in the **Case name** field. This facilitates the management of the OpenFoam remote and local cases.

Activate the **Execute case** button to send the CFD case to the OpenFoam interface.

### 5.2.3 Create CFD Case from a Volume Mesh

Start the CFD simulation activating the **Create CFD case** green button. A dialog window appears as illustrated below:
Toggle the **Load existing volume mesh** radio button to start the creation of the CFD case from a VM. Enable the **from local cases manager** radio button. The local cases are listed in a dialog window as illustrated below:

The left side shows the list of the available local cases. The list allows selecting one of the volume meshes for simulation. With the **Case Preview** box enabled, the selected VM is shown in the image port. The general image manipulation panel is available on the right hand side. It allows changing the appearance of the image in the viewport. The volume rendering displayed in the viewport can be saved as an image with the **Save** button.

With the **Load image data** box enabled it is possible to **Set paths** for velocity and magnitude image data. Close the dialog window with the **Set** button to confirm the images settings.

Activate the **OK** button to exit the window and continue with the creation of the CFD case. The interface is simplified as shown below:
It consists of the **Boundary Conditions** and the **Solution Controls** panels. Define the **Boundary Conditions** as explained previously. Finally, define the parameters for the solver. Set the number of **iterations** to a full tens number, e.g. 80 and enable the **Get final results only** box to save only the final solution for the simulation.

**Note:**

- Assign a meaningful name to the CFD case in the **Case name** field. This facilitates the management of the OpenFoam remote and local cases.

- The **View volume mesh** button allows viewing the VM in the 3D interface and interactively explore it.

- Send the CFD case to the OpenFoam interface by activating the **Execute case** button.

### 5.2.4 Create CFD case from an Existing Case

Start the CFD simulation activating the **Create CFD case** green button. A dialog window appears as illustrated below:
Enable the **Load case** button to start the CFD simulation from a previously saved case. Close the window with the **OK** and select a `.cfdCase` file. The CFD interface appears displaying the **Volume Mesh** as the first step of the analysis.

The `.cfdCase` file contains all the settings necessary for the CFD simulation: **Volume Mesh**, **Boundary Conditions** and **Solution Controls**. Modify the case definitions (e.g. **Boundary Conditions**) in order to try variations of the processing parameters. Save the new case settings with the **Save** button and finally send the new CFD case to the server with the **Execute case** button.

**Note:**

Assign a meaningful name to the CFD case in the **Case name** field. This facilitates the management of the OpenFoam remote and local cases.

The **View surface mesh** button allows viewing the SM in the 3D interface and interactively explore it.

### 5.2.5 Management of the OpenFoam Remote Cases

The Open Foam remote cases are easily accessible via the dedicated button available in the lateral task bar. A dialog window opens as illustrated below:
The information related with the OpenFoam server is displayed in the upper section.

The cases available on the server are centrally listed in table format. The status of each case is displayed on the **Status** column: it can be either **running**, **failed** (giving number of errors), or just ticked ✓ if successfully finished as in the capture above.

The functionality of the action buttons available at the bottom of the page are summarized in the table below:

<table>
<thead>
<tr>
<th>Button</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Refresh" /></td>
<td>Allows refreshing the case list</td>
</tr>
<tr>
<td><img src="image" alt="Stop" /></td>
<td>Allows stopping a running case on the list.</td>
</tr>
<tr>
<td><img src="image" alt="Logs" /></td>
<td>Allows visualizing the logs file of the cases in the list. Its activation opens a window like illustrated below:</td>
</tr>
</tbody>
</table>

The window is organized on pages which content can be **Copy to Clipboard** and paste in a text file.

| ![Errors](image) | Allows visualizing the error logs. Its activation opens a dialog window as shown below: |
The window is organized on pages which content can be Copy to Clipboard and paste in a text file.

Residuals

Allows visualizing the residuals for the case selected in the list. A dialog window opens and shows the plotted residuals as illustrated below:

In this example the residual becomes flat after approximately 30 iterations meaning that solution is converged.

Delete

Allows deleting the selected case from the list

Download

Allows downloading the selected case from the list. Optionally, the case can be deleted from the OpenFOAM server enabling the Delete checkbox aside the Download button.

When activated a confirmation dialog appears as illustrated below:

It shows the location where the file will be saved. Please note that this setting is configured locally within the PGEM interface as explained above.

The AortaCase was selected for download.
5.2.6 Management of the OpenFoam Local Cases

Activate the Manage OpenFOAM Local cases button from the lateral task bar. The local cases dialog window is illustrated below:

The left side shows the list of the available local cases. The list allows selecting one of the local cases for visualization and inspection in the 3D page. With the Case Preview box enabled, the result of the selected case is shown in the image port. The general image manipulation panel is available on the right hand side. It allows changing the appearance of the image in the viewport. The CFD results previewed in the viewport can be converted into images with the specified Resolution using the Generate button in the Image data result panel. The images series are saved in the specified format using the Save button.

The functionality of the action buttons available under the local cases list are summarized in the table below:

<table>
<thead>
<tr>
<th>Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>🔄 Refresh</td>
<td>Allows refreshing the case list</td>
</tr>
<tr>
<td>❌ Stop</td>
<td>Allows stopping a running case on the list</td>
</tr>
<tr>
<td>📡 Logs</td>
<td>Allows visualizing the logs file of the cases in the list. Its activation opens a window like illustrated below:</td>
</tr>
</tbody>
</table>
The window is organized on pages which content can be **Copy to Clipboard** and paste in a text file.

### Errors

Allows visualizing the error logs. Its activation opens a dialog window as shown below:

![Error Log](image)

The window is organized on pages which content can be **Copy to Clipboard** and paste in a text file.

### Residuals

Allows visualizing the residuals for the case selected in the list. A dialog window opens and shows the plotted residuals as illustrated below:

![Residuals Plot](image)

In this example the residual becomes flat after approximately 30 iterations meaning that solution is converged.

### Delete

Allows deleting the selected case from the list.

### Move

Allows moving selected cases to the back-up folder (path set in PGEM configuration as explain above). The back-up folder is not scanned, therefore the local cases dialog will open up faster.

When activated a dialog appears as illustrated below:

![Move Dialog](image)
5.2.6.1 Conversion of CFD Results to Images

The Generate button in the Image data results panel allows creating image data with the specified Resolution.
Four image series are created: Geometry, p (Pressure), U (Velocity magnitude), U (Velocity XYZ components).

Use the Save icon to save the currently shown series in any of the supported formats. If the All box is checked, all image series are saved in separate files with the specified resolution.

5.2.6.2 Visualization of the CFD Results

The View button allows displaying the results of the CFD case selected in the list in a 3D window. There are two options for visualization of the CFD results: as Polymesh or as Volume rendering:

5.2.6.2.1 As Polymesh

Select the Volume mesh (Polymesh) option in the list and activate the View button to transfer the CFD results to the 3D page. An intermediate dialog window allows selecting the original 4D Flow data:

The selected image series may be used for texturing in the 3D interface only if the same data were used to create the geometric model used for the simulation.

The result for a Brain Aneurysm data is illustrated below:
For further visualisation, an oblique cutting plane is applied to the aneurysm CFD results, to reveal internal values for velocity and pressure:
The following settings were used for visualization:

1. All the objects in the **View** tree were set to **Solid**.
2. The **wallsPatchVisible** box was switched OFF.
3. An oblique plane was used to cut the **cells_boundary**. After cutting, the plane **Visible** box was set to OFF.
4. The **p** (pressure) and the **U** (velocity) **Magnitude** selected on **Frame 2** were used to texture the **cells_boundary** object.

The resulting velocity field and pressure can be compared visually to the known velocity from 4D Flow. To this purpose the loaded **image data** with **XYZ_velocity** selection was overlaid on the **cells_boundary** object:
The wall shear stress maps can also be assessed in the 3D environment. For this purpose the `wallsPatchVisible` box is enabled and the `wallShearStressMagnitudeFrame 2` selected as texture. The scene is updated as illustrated in the capture below:
5.2.6.2.2 As Volume Rendering

Visualization of the CFD Results as Volume Rendering

With the **Volume rendering** selection additional settings are available, as illustrated below:

The settings are summarized in the following table:

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Resolution</td>
<td>Allows setting the image resolution. The following selection is available:</td>
</tr>
<tr>
<td>128</td>
<td>64, 128, 256 and 512.</td>
</tr>
</tbody>
</table>

The selected number indicates how many cubes will be rendered in each direction. Each cube stores the information about the variable field from one cell of the volume mesh. The higher the resolution the better the precision, but the memory requirements grow significantly.

The rendering results for two different resolutions (128 and 512) are illustrated below for the velocity magnitude estimated for the Aorta model:

**Note:**

In the 3D scene the cubes will be drawn in the positions which represent the centers of the volume mesh cells.

The following settings were used for visualization:

1. The resolution was set to 512.

The simulation results are transferred to the 3D page activating the **View** button. They are available as images on the **Input** tab of the 3D page. In the **View** tree, the **CFD_Geometry** is available as the volume render object, the **CFD_p_(Pressure)** and **CFD_V_(Velocity_magnitude)** as images for the texture.

The two physical variables, velocity and pressure, are visualized as textures on the same geometry as illustrated below:
In addition, plane elements can be defined for composing a meaningful scene. The planes serve for defining the parts to be cut out of a VR object.

Please refer to the *PMOD 3D Rendering Tool Users Guide* for any further information about the rendering.
6 References

7 PMOD Copyright Notice

Copyright © 1996-2021 PMOD Technologies LLC. All rights reserved.

The PMOD software contains proprietary information of PMOD Technologies LLC; it is provided under a license agreement containing restrictions on use and disclosure and is also protected by copyright law. Reverse engineering of the software is prohibited.

Due to continued product development the program may change and no longer exactly correspond to this document. The information and intellectual property contained herein is confidential between PMOD Technologies LLC and the client and remains the exclusive property of PMOD Technologies LLC. If you find any problems in the document, please report them to us in writing. PMOD Technologies LLC does not warrant that this document is error-free.

No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording or otherwise without the prior written permission of PMOD Technologies LLC.

PMOD Technologies LLC
Sumatrastrasse 25
8006 Zürich
Switzerland
+41 (44) 350 46 00
support@pmod.com
http://www.pmod.com
This page is intentionally left blank.
<table>
<thead>
<tr>
<th>Index</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4D Flow</td>
<td>28, 72</td>
</tr>
<tr>
<td>4D Flow Report</td>
<td>37</td>
</tr>
<tr>
<td>A</td>
<td></td>
</tr>
<tr>
<td>Additional Documentation</td>
<td>13</td>
</tr>
<tr>
<td>Anatomical Model</td>
<td>46, 60</td>
</tr>
<tr>
<td>Anatomical Model Definition</td>
<td>43</td>
</tr>
<tr>
<td>As Polymesh</td>
<td>95</td>
</tr>
<tr>
<td>As Volume Rendering</td>
<td>99</td>
</tr>
<tr>
<td>B</td>
<td></td>
</tr>
<tr>
<td>Boundary Condition Settings</td>
<td>82, 85, 86</td>
</tr>
<tr>
<td>C</td>
<td></td>
</tr>
<tr>
<td>Calculating the Fiber Tracks</td>
<td>18</td>
</tr>
<tr>
<td>Calculating the Streamlines</td>
<td>33</td>
</tr>
<tr>
<td>CFD Simulation and Implementation in PGEM</td>
<td>72</td>
</tr>
<tr>
<td>Components of the Model Folder in PMOD</td>
<td>64</td>
</tr>
<tr>
<td>Configuration Settings</td>
<td>8, 89, 92</td>
</tr>
<tr>
<td>Conversion of CFD Results to Images</td>
<td>94</td>
</tr>
<tr>
<td>Create CFD Case from a Surface Mesh</td>
<td>85</td>
</tr>
<tr>
<td>Create CFD Case from a Volume Mesh</td>
<td>86</td>
</tr>
<tr>
<td>Create CFD case from an Existing Case</td>
<td>88</td>
</tr>
<tr>
<td>Create CFD Case from the Geometric Model</td>
<td>74</td>
</tr>
<tr>
<td>Creating VOIs along the Streamlines</td>
<td>40</td>
</tr>
<tr>
<td>Cutting Planes Definition with 3D</td>
<td>77</td>
</tr>
<tr>
<td>D</td>
<td></td>
</tr>
<tr>
<td>Defining VOIs for Seeding and Imposing Restrictions</td>
<td>15, 30</td>
</tr>
<tr>
<td>Definition of the Characteristics for the Model's Elements</td>
<td>61, 64, 67</td>
</tr>
<tr>
<td>Definition of the Properties for the Model's Elements</td>
<td>69</td>
</tr>
<tr>
<td>DTI Report</td>
<td>24</td>
</tr>
<tr>
<td>DTI Tractography</td>
<td>14, 28</td>
</tr>
<tr>
<td>Dynamic PET Image Creation</td>
<td>43</td>
</tr>
<tr>
<td>E</td>
<td></td>
</tr>
<tr>
<td>Entering the Maximal Flow Value</td>
<td>30</td>
</tr>
<tr>
<td>Evaluating Diffusion Images along Tracks</td>
<td>26</td>
</tr>
<tr>
<td>External Preparations</td>
<td>42</td>
</tr>
<tr>
<td>G</td>
<td></td>
</tr>
<tr>
<td>Generation of Dynamic PET Phantom Images</td>
<td>42, 49</td>
</tr>
<tr>
<td>Geometric Model Construction</td>
<td>49</td>
</tr>
<tr>
<td>Geometric Model Construction and CFD Simulation</td>
<td>49</td>
</tr>
<tr>
<td>Geometry Selection in the Model</td>
<td>74</td>
</tr>
<tr>
<td>I</td>
<td></td>
</tr>
<tr>
<td>Image Definition</td>
<td>52</td>
</tr>
<tr>
<td>Image Loading</td>
<td>5</td>
</tr>
<tr>
<td>Interactive Exploration of the Model</td>
<td>70</td>
</tr>
<tr>
<td>L</td>
<td></td>
</tr>
<tr>
<td>Loading the 4D Flow Images</td>
<td>28</td>
</tr>
<tr>
<td>Loading the Diffusion Tensor Images</td>
<td>14</td>
</tr>
<tr>
<td>M</td>
<td></td>
</tr>
<tr>
<td>Management of the OpenFoam Local Cases</td>
<td>92</td>
</tr>
<tr>
<td>Management of the OpenFoam Remote Cases</td>
<td>89</td>
</tr>
<tr>
<td>O</td>
<td></td>
</tr>
<tr>
<td>OpenFoam CFD Server</td>
<td>8, 9</td>
</tr>
<tr>
<td>P</td>
<td></td>
</tr>
<tr>
<td>Phantom Creation</td>
<td>46</td>
</tr>
<tr>
<td>Phantom Dynamic TACs</td>
<td>42</td>
</tr>
<tr>
<td>Phantom VOIs</td>
<td>42</td>
</tr>
<tr>
<td>PMOD Anatomy Tool (PGEM)</td>
<td>5</td>
</tr>
<tr>
<td>PMOD Copyright Notice</td>
<td>103</td>
</tr>
<tr>
<td>Protocol File</td>
<td>27, 41</td>
</tr>
<tr>
<td>R</td>
<td></td>
</tr>
<tr>
<td>References</td>
<td>102</td>
</tr>
<tr>
<td>S</td>
<td></td>
</tr>
<tr>
<td>Saving the Streamlines</td>
<td>41</td>
</tr>
<tr>
<td>Saving the Tracks</td>
<td>26</td>
</tr>
<tr>
<td>Solution Control</td>
<td>84</td>
</tr>
<tr>
<td>Start of the Model Definition</td>
<td>49</td>
</tr>
<tr>
<td>Surface Mesh Creation</td>
<td>75</td>
</tr>
</tbody>
</table>
- T -
Time Definition 50, 55
Tree Organization of the Anatomical Model 61

- U -
User Interface 5

- V -
Visualization of the CFD Results 95
Visualizing the Fiber Tracks in 3D 21
Visualizing the Streamlines in 3D 36
VOI Addition 45, 56, 61, 74
VOI Definition 55
Volume Mesh Creation 80

- W -
Workflow Organization 14, 28, 49